



POLITECNICO
MILANO 1863

Course of "Fluid Labs" A.A. 2024-2025

CFD Test Case 1 Laminar oil transport in a pipeline

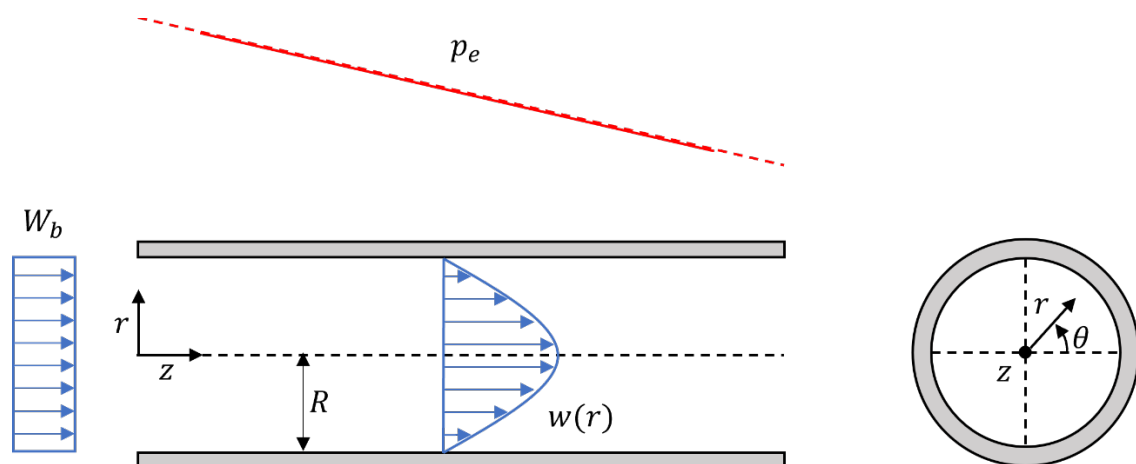


Figure 1. Sketch of the case

Case description

The first test case is the laminar flow of oil in a pipeline (Figure 1). If the pipeline is sufficiently long, one expects the flow to be fully developed, and this state must be reproduced in the CFD solution. However, if a uniform velocity profile is imposed at the inlet boundary, the fully developed state is reached at a certain distance downstream of it. The fully developed, laminar flow in a pipe admits an analytical solution, which is the cylindrical Poiseuille flow:

$$w(r, \theta, z) = w(z) = -\frac{1}{4\mu} \frac{dp_e}{dz} (R^2 - r^2) \quad u(r, \theta, z) = 0 \quad v(r, \theta, z) = 0$$

$$\frac{dp_e}{dz} = \text{const} < 0$$

$$\tau_{rz}(r, \theta, z) = -2\mu \frac{1}{2} \left(\frac{\partial v}{\partial z} + \frac{\partial w}{\partial r} \right) = -\mu \frac{dw}{dr} = -\frac{dp_e}{dz} \frac{r}{2}$$

where μ is the dynamic viscosity of the fluid, R is the inner radius of the pipe, u, v, w are the velocity components along directions θ, r, z (Figure 1 and Figure 2), p_e is the excess pressure (that is, the part of pressure which is not balance by gravity), and τ_{rz} is the only nonzero shear stress term.

Numerical simulation

The steady-state formulation of the Navier-Stokes equations are solved. The applied boundary conditions are inlet, outlet, wall and axis¹. The computational burden of the simulations can be reduced by exploiting the axisymmetry of the problem, which will be therefore simulated as 2D through a domain having the shape of a circular sector, as shown in Figure 2. At the inlet, a uniform z -velocity profile equal to W_b (bulk velocity) is imposed, whereas the r - and θ velocity components are set to zero. At the outlet, the excess pressure is specified to a given value, which might be zero. The walls are assumed fixed and a no-slip condition is set at the boundaries, the fluid velocity is zero at the walls. Being in the laminar case, the wall shear stress is simply evaluated as the axial velocity in the near-wall cells divided by its distance from the wall and multiplied by the fluid viscosity. This set of boundary conditions results in a developing flow field which, starting from the uniform distribution imposed at the inlet section, reaches a fully developed configuration at a certain distance downstream of it.

Configuration of the problem

Pipe diameter = 150 mm,

Bulk velocity $W_b = 0.45$ m/s,

Fluid = heavy crude oil with density $\rho = 910$ kg/m³ and kinematic viscosity $\nu = 3.5 \cdot 10^{-4}$ m²/s

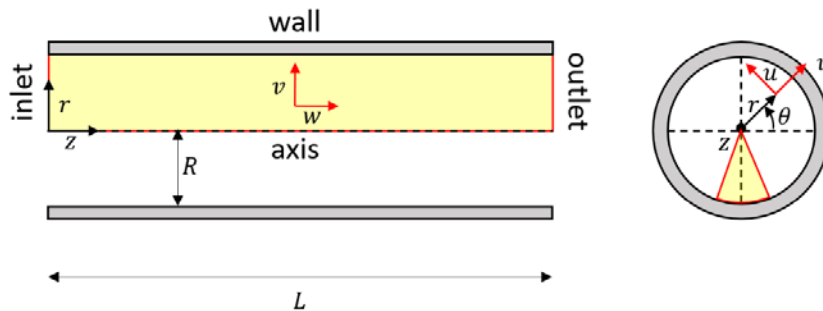


Figure 2. Domain and boundary conditions

¹ Note that other combinations of boundary conditions are possible for this problem. These include, for instance: (i) outlet (used as inflow), outlet, wall, and axis and (ii) periodic boundary conditions with imposed mass flow rate, wall, and axis.

Questions

Simulate the flow in PHOENICS, addressing the following issues.

1. Verify that the **testing conditions indicated correspond to laminar flow**. This is achieved by calculating the bulk Reynolds number and Re_b , comparing it against the threshold value of about 2000.
2. Set the **domain length, L , in such a way that fully developed flow conditions can be attained from the uniform velocity profile imposed at the inlet**. Note that the goal is not to determine accurately the length required for flow development (often called “entry length”), but just **to observe fully developed flow in the CFD solution**. Note also that, in principle, fully developed flow is reached in an asymptotic manner, that is, at an infinite distance downstream of the inlet. Hence, you must develop yourself a practical criterion should be established to define what fully developed flow is; any criterion has strengths and weaknesses, so try to highlight them.

*Hint: do not belittle the **qualitative impact**! Sometimes a qualitative criterion is more effective than a too abstract quantitative one.*

*Hint: many **formulas have been reported in the literature** for estimating the entry length of laminar pipe flow. Try to find some and compare against your CFD results. Pay attention to check whether the comparison is consistent: for instance, is the meaning of the “entry length” in the literature formulas the same that you are attributing to this parameter in your calculations? Also, if you can find many formulas providing different estimates, do not limit to those providing the best match with your calculations; instead, ask yourself “why do these formulas provide different estimates for the same parameters? Why do some formulas agree with my CFD results and others do not?”*

3. **Verify whether the CFD solution is physically sound** from a qualitative point of view, that is, check whether there is “anything strange” in the results when **inspecting the colour plots**.
4. **Assess the numerical convergence of the CFD solution**. This requires, on the one hand, verifying that the **CFD simulation must be converged** with respect to the solution algorithm and, on the other, that the **solution is grid independent**. The convergence with respect to the algorithm is achieved through the monitoring of the normalized whole-field residuals and probe values, setting up an adequate maximum number of iterations and a suitable global convergence criterion. The grid-independence study must be carried out based on suitable target parameters. Note that the target parameters are decided by the user, and all quantities subject of investigation must be proven grid-independent.

*Hint: **how do you design the mesh pattern** for this case? Why? Once you have defined the first mesh, how do you define the (at least) two others to carry out the grid independence study?*

5. **Validate the CFD solution in the fully developed region with respect to the analytical solution of Poiseuille flow**. Since there exists an analytical solution for this case, this validation is called **benchmarking**, and its purpose is simply to self-verify the correct implementation of the case, as well as the numerical convergence. The validation of the CFD solution must be made in terms of the velocity $w(r)$ and the shear stress $\tau_{rz}(r)$ profiles, as well as in terms of the (excess) pressure gradient dp_e/dz .

Supplementary question (optional)

6. Manipulate the analytical solution to **obtain the trend of the friction factor f versus the bulk Reynolds number, Re_b** , which is $f = 64/Re_b$. Compare the CFD results and the analytical solution in terms of f for the present case study. **Extend the comparison to other values of Re_b** (that is, for different velocities or different pipe diameters). Note that, for all cases, the flow must be laminar, thus having Re_b **smaller than say 2000**.

7. In the academic and professional practice, formal techniques for estimating the grid discretization errors are sometimes used, and one of the most well known is the Grid Convergence Index (GCI) method by Roache. Such formal techniques are useful but they are not free from criticisms, including the difficulties in evaluating some input parameters and coefficients for complex cases. As an attachment, you are given a short paper by Ismail Celik from West Virginia University, in which a formal, systematic procedure for evaluating the grid discretization error is given, which generalizes the GCI method by Roache. If you are interested, employ this procedure for strengthening your grid independence assessment.

Hint: do not limit yourself to apply the formal procedure and obtain the uncertainty estimates, but try to assess its strenghts and, most important, its weaknesses.